SCHEMA III Quick Reference Guide

INSTALLATION ____ 1

MANAGER ____ 2

DRAWING ____ 3

CREATE A PART ____ 4

POST-PROCESSING & HIERARCHY ____ 5

LIBRARIES ____ 6

CONFIG FILES ____ 7

PRINTING ____ 8

PLOTTING ____ 9

PAGES ____ 10

THIRD PARTY CONNECTIVITY ____ 11

ECO TO SCHEMA-PCB ____ 12

OVERVIEW

SCHEMA is a powerful and flexible Schematic Capture program. Through the years, it has evolved into a multi-faceted software package with many interfaces. However, flexibilty breeds complexity, and that is reason for this *Quick Reference Guide*. If you are a new user of **SCHEMA** we suggest that you complete the Tutorial, Chapter 1 / *Getting Started,* as minimum requirement. Then read the *SCHEMA III Reference Manual* in its entirety at least once. We hope this *Quick Reference Guide* will aide in your understanding of **SCHEMA** and its capabilities.

INSTALLATION

1. Place PROGRAM DISK 1 into drive A: and type: **INSTALL <Enter>.**

2. The Main Installation Menu appears with the following items :

SCHEMA DIRECTORY - Change the drive and path to where you wish to install the SCHEMA software.

LIBRARY DIRECTORY - Change the drive and path to where you wish to install the SCHEMA Object Libraries.

GRAPHICS ADAPTER - Select from the list of adapters the card you will be using.

PRINTER DRIVER - Select from the list of printer drivers the printer you will be using.

PEN PLOTTER SETUP - Select from the list of plotter drivers the plotter you will be using. The choices are: Houston Instruments, Hewlett Packard, Calcomp, DXF, and PostScript.

INSTALL SCHEMA III SOFTWARE - Select Yes if you want install the SCHEMA III Software.

INSTALL SCHEMA-PCB INTERFACE - Select Yes if you wish to install the SCHEMA-PCB interface. The interface consists of the additional executables for the Annotation process as well as adding the SCHEMA-PCB commands to column 3 of the SCHEMA Program Manager. When the Installation is finished if this option is YES then you will be prompted to enter the Main Disk 1 of your SCHEMA-PCB Software. If you do not wish to install SCHEMA-PCB then press **<Esc>**.

INSTALL WITH ABOVE SELECTIONS - Selecting this command will begin the Installation Process.

EXIT INSTALL - This command exits the Install.

Note: This Install is non-destructive. If you already have SCHEMA installed on your system then the Install Program will read your current SCHEMA.CFG to setup the Main Install Screen. The old SCHEMA.CFG is backed up to the file SCHEMA.CCC and the SCHEMA.MGR is backed up to the file SCHEMA.MMM.

The Install Program will alter the CONFIG.SYS and the AUTOEXEC.BAT. The following statements are added:

CONFIG.SYS - **FILES=20 BUFFERS=20**

AUTOEXEC.BAT - **OMATION=C:\SCHEMA** *(Environment Variable)*

PATH=C:\SCHEMA

Reference: See *APPENDIX A.2 & A.3* in the *SCHEMA III Reference Manual* for more information about the installation of the software

SCHEMA MANAGER

The SCHEMA Program Manager is a user-definable system manager that gives the user the integration of all SCHEMA products as well as any other software products that the user may wish to add. The manager can be configured on-line using the **EDIT** command. This manager uses the double click method of operation. The first mouse click or **<Enter>** being pressed puts the cursor at the Command Line. The second performs the execution of the selected Manager command.

Column 1 contains CONTROL or MANAGER specific commands. They are :

HELP - Invokes the On-line Help Mode. Select the command and the and if a file with the command prefix with a .HLP extension is found then the file will be displayed for viewing. See the VIEW command for more information.

QUIT - Quits the Manager to DOS .

SAVE - Saves the current configuration of the manager to SCHEMA.MGR file, and backs up the previous file to SCHEMA.BAK.

FILTER - Allows you to change the filter. (Example: *.P01)

PATH - Used to alter the command path.

FILE - Used to alter the current file.

EDIT - Invokes the Edit Mode of the manager to customize its operation to the user's specifications. When selected the Manager goes into the Edit Mode. Select the command slot that you wish to alter. Be sure to press **<Enter>** for every line that you change. When finished altering a command slot press the **<Up Arrow>** to leave the edit window. Exit the Edit Mode by pressing **<Esc>** or the right mouse button. Select **SAVE** to store the changes just made to the Manager.

VIEW - Allows the user to view a text file. Include the full path to the file if it does not reside in the current directory. The left mouse button performs the same operation as **<Pg Dn>** . The right mouse button is used to **<Esc>** from viewing the file. For those who have a three button mouse the middle button performs the **<Pg Up>** operation.

Column 2 contains the **SCHEMA** Commands. They are:

NEW_PAGE - Creates a new schematic page from template files. When selected a list of template pages with the extensions .P99 appear. An explanation of the different pages is found later in this manual in the *PAGES* section. Select a template file and you are prompted to whether you want to create a new drawing page. Answer 'Y'. Then you are prompted for the drawing file: path, name, and extension. Once given the prompt, "**CREATE NEW CONFIGURATION FILE (Y/N)**" appears. This is your choice to whether you want a config file for this project. A 'N' answer will cause the **SCHEMA.CFG** file to be used as default.

DRAWINGS - Used to select an existing schematic page.

SCHDRAW - Invokes the Schematic / Object Editor.

SCHPOST - Invokes the Post-processing for the schematic.

SCHPRINT - Prints the schematic pages.

SCHPLOT - Plots the schematic pages.

SCHCFG - Invoke either drawing .CFG files or **SCHEMA.CFG**.

SCHLIB - Invokes the Library Manager.

SCHLOCAL - Localizes all the parts in a set of schematics into one library.

SCHBACK - Used for Back Annotating to the schematics. If using **SCHEMA-PCB, SCHECO** will invoke **SCHBACK** automatically.

SCHPAGE - Creates custom schematic pages.

Column 3 may contain the **SCHEMA-PCB** commands. These are only created for the MANAGER, if the Installation for **SCHEMA** with the **SCHEMA-PCB** is chosen. These commands are:

SCHPCB - Invokes the **SCHEMA-PCB** program.

SLIB - Invokes the **SCHEMA-PCB** Library Manager.

SEDIT - Invokes the ASCII Text editor.

SCHECO - Invokes the Forward or Back Annotator.

S_ROUTER - Invokes the **SCHEMA-SUPERROUTER** Option.

SREAD - Reads a .JOB to a **SCHEMA-SUPERROUTER** file.

SWRITE - Writes a **SCHEMA-SUPERROUTER** file to a .JOB.

2

Column 7 contains SYSTEM or DOS commands.

Reference: See Chapter 2, *SCHEMA Program Manager* in the *SCHEMA III Reference Manual*

SCHDRAW MOUSE / KEYBOARD USAGE

MOUSE

< Left Button > - Use to select a command or do an editing function.

< Middle Button > - Activates the Command Window indicator to scroll for command selection.

< Right Button > - Undo last command.

KEYBOARD

TEXT EDITING KEYS

< Esc > - Clears the Text Window.

< Backspace > - Erases one character to the left and shifts the characters.

< Del > - Erases the current cursor character, but does not shift characters.

< Left Arrow > - Moves the cursor left one position.

< Right Arrow > - Moves the cursor right one position.

< Home > - Moves the character to the first position.

< End > - Moves the cursor to last position.

SCHDRAW KEYS

(*Use the Numeric Keypad with Num Lock turned OFF*)

< Arrow Keys > - Moves the cursor up, down, left, or right.

< Ctrl > < Left Arrow > - Jumps the cursor to the left edge of the drawing.

< Ctrl > < Right Arrow > - Moves the curser to the right edge of the drawing.

< PgDn > - Jumps the cursor to the bottom edge of the drawing.

<PgUp> - Jumps the cursor to the top edge of the drawing.

<Home> - Jumps the cursor to the center of the drawing.

<F1> - Next keypress is the first letter of the Primary Command.

<F2> - Next keypress is the first letter of the Secondary Command.

<F3> - Redisplays the last text string in the Text Window.

<F4> - Turns the **CONFIG-DIAG** command on/off for diagonal lines.

<F5> - Turns the Cursor Coordinate Window on/off.

<F7> - ZOOMs IN one level.

<F8> - ZOOMs OUT one level.

<F9> - Repeats the previous command. Can be used in the Schematic Editor in place of the **DRAW-REPEAT** command, or in the Object Editor to repeat pin headers and definitions.

<F10> - Turns the Cursor Grid (C GRID) on/off.

<Ins> - Simulates pressing the left mouse button to select a command or do an editing function. Pressing this key again, simulates releasing the left mouse button.

<Esc> - Activates the **UNDO** command sequence.

PARTS BROWSING KEYS

(*Use the Numeric Keypad with Num Lock turned OFF*)

<Home> - Moves the cursor to the first part on the screen.

<PgUp> - Moves the cursor to the previous screen of parts.

<PgDn> - Moves the cursor to the next screen of parts.

<End> - Moves the cursor to the last part on the screen.

<Arrow Keys> - Moves the part cursor up, down, left, or right.

<Ins> - Displays the part at the cursor. Pressing **<Ins>** again returns to the listing of the parts.

SCHDRAW

COMMANDS

BODY : Used to create Body Objects.

BODY-BIT : To create bit images pixel by pixel.

BODY-CIRCLE : Add circles and arcs to objects.

BODY-DF PI : Assign pin definitions to pin headers of an object.

BODY-HDR : Add pin headers(stubs) to objects.

BODY-LINE : Add thin, thick, dotted, or dashed lines to objects.

BODY-N OBJ : Add a NESTED Object to a current Body/Nested Object.

BODY-PARM : Assign parameters to an object.

BODY-TEXT : Add small, medium, or large text to an object .

CONFIG : Set ON/OFF **SCHDRAW** configuration parameters.

CONFIG-COLORS: Alter the Schematic Editor colors On-line.

CONFIG-C GRID : Turn ON/OFF cursor grid.

CONFIG-DIAG : Turn ON/OFF diagonal line capabilities.

CONFIG-LIBR : Display the currently loaded libraries.

CONFIG-PAN X/Y: Adjust the panning speed 0-Fast thru 19-Slow.

CONFIG-RPT X/Y: Set the repeat offsets for X and Y.

CONFIG-V GRID : Turn ON/OFF the Visual grid.

CONFIG-WI HDR : Turn ON/OFF wire, pin, and label headers.

CONFIG-XY WDW : Turn ON/OFF the Cursor Coordinate Window.

DRAW : Add graphic objects: objects, wires, dots, arcs, etc.

DRAW-BUS : Add thin or thick 90 degree bus arcs.

DRAW-DOT : Add a junction dot to three meeting wires.

DRAW-GATE : Increments to the next available gate section.

DRAW-LABEL : Label a net on the schematic.

DRAW-MOVE : Move an object on the schematic page.

DRAW-OBJECT : Add an object from a library to the schematic.

DRAW-REPEAT : To step and repeat **WIRE, ARC, OBJECT,** and **LABEL.**

DRAW-ROTATE : Increments to the next available rotated version.

DRAW-WIRE : Draw orthogonal thin and thick wires for signal or bus.

EDIT : To edit existing graphics.

EDIT-COPY : To copy an image or region in the schematic.

EDIT-ERASE : To erase an image or region in the schematic.

EDIT-GATE : Increments to the next available gate section.

EDIT-ID : To identify Name/Body/Nested Object subcomponents.

EDIT-IMAGE : To edit the **DRAW-OBJECT** line or move wires.

EDIT-LOAD : Used in conjunction with **EDIT-REGION** to load a region.

EDIT-MOVE : To move an object or a region.

EDIT-NEWREF : To maintain or change REFDs when **COPY/LOAD** a region.

EDIT-PM OBJ : Display Body Object PARMs.

EDIT-REGION : Define a region to **MOVE, COPY, LOAD, ERASE,** or **SAVE.** Also can be used to read in a Ascii Text file.

EDIT-ROTATE : Increments to the next available rotated version.

EDIT-SAVE : Save a defined region to disk.

EDIT-SELECT : To select an image to **MOVE, COPY, ERASE,** or **EDIT.**

EDIT-UPDATE : To edit pin definitions **(BODY-DF PI)** or **BODY-TEXT.**

FILE : To load, save, redraw, merge, or print a schematic.

FILE-ED OBJ : To invoke the Object Editor.

FILE-LOAD : Loads a schematic page into the Drawing Window.

FILE-POP NN : To select a file off the **PUSH/POP** stack for hierarchy.

FILE-PRINT : Prints the schematic in foreground or background mode.

FILE-PUSH : To put a file on the file stack of hierarchy or load non-electrical or text files into the Schematic Editor.

FILE-QUIT : To exit **SCHDRAW** and return to DOS or the Manager.

FILE-REDRAW : To redraw the graphics of the current drawing.

FILE-SAVE : To save the current drawing to disk.

HELP : To invoke the on-line help mode of **SCHDRAW.**

LIBR : To load, save, or redraw an object.

LIBR-ED SCH : To invoke the Schematic Editor.

LIBR-LOAD : To load a Name/Body/Nested Object into the Object Editor.

LIBR-NEW : To clear the Object Editor Window.

LIBR-QUIT : To exit **SCHDRAW** and return to DOS or the Manager.

LIBR-REDRAW : To redraw the Object Editor Window.

LIBR-SAVE : To save a Name/Body/Nested Object to a library on disk.

NAME : To create and edit NAME Objects.

NAME-BODY : To assign the Body field to the NAME Object.

NAME-DESC : To assign description fields 1 - 4 to the NAME Object.

NOTES : To add non schematic graphics to the schematic.

NOTES-CIRCLE : To add circles or arcs to the schematic.

NOTES-LINE : To add circle/arcs, thin, thick, dotted, and dashed lines.

NOTES-TEXT : Add small, medium, and large text to the schematic.

SEARCH : To search for a coordinate, object, REFD, label, and text.

SEARCH-LABEL : To locate a particular label.

SEARCH-NAME : To locate a particular object.

SEARCH-REFD : To locate a particular reference designator.

SEARCH-TEXT : To locate a particular string of text.

SEARCH-X,Y : To locate a particular coordinate in the drawing.

UNDO : This command is no longer available as a menu item but the <Esc> key or the right mouse button perform the same operation. Its purpose is to undo the last command executed.

ZOOM : To view and edit in four zoom levels. (1:1, 2:1, 4:1, AND 8:1)

BROWSE

Online parts browsing can be executed from the **DRAW-OBJECT** command in the Schematic Editor or **LIBR-LOAD** in the Object Editor.

I. Select either **DRAW-OBJECT** or **LIBR-LOAD**.

II. Then you can type the following string into the command text window:

*** <Enter>** (Displays all the objects in LIBR1)

n=* <Enter> (Displays all the objects in LIBRn, where n is the number 1 through 9)

ab* <Enter> (Displays all objects that begin with "ab" in LIBR1)

n=ab* (Displays all objects that begin with ab, in LIBRn, where n is the number 1 through 9)

III. After this command string is entered the Directory Screen appears with the object names in alphanumeric order, left to right.

IV. The following keys are recognized in Browse :

MOUSE

<Left Button> - The first click selects the item to view. The second click selects the item and displays it on the **DRAW-OBJECT** Parameter string.

<Right Button> - This button with either exit from viewing a part and return to the browse listing or exit browse and return to the Schematic Editor.

Keyboard *(Numeric Keypad only)*

<PgUp> - Displays the previous screen of objects.

<PgDn> - Displays the next screen of objects.

<Home> - Moves the cursor to the first object in the screen.

<End> - Moves the cursor to the last object in the screen.

<Ins> - Pressing once displays the part. Use the arrow keys if you need to pan to see the whole part. Press **<Ins>** a second time to return to the directory.

<Arrow keys> - Moves the cursor or scroll an object.

<Esc> - Returns to the directory screen or to the Text Window.

<Enter> - Selects the object and returns it to the Text Window.

ORIENTATION - DeMORGAN / ROTATE

The most of gated parts of the standard libraries are supplied with rotated orientation of gates, gate sections and DeMorgan equivalents. For example an AND gate, T08-1A is the normal orientation where T08-2A is the DeMorgan equivalent.

There are two ways in which to invoke the gated, rotated or DeMorgan equivalents.

The first is to use the **DRAW-GATE** and **DRAW-ROTATE** commands. When selected these commands increment to the next available section if it exists. **EDIT-GATE** and **EDIT-ROTATE** are equivalent commands.

The second method is to specify the sections directly on the **DRAW-OBJECT** Parameter Line.

1 - Normal (left-to-right) *default*

2 - DeMorgan (left -to-right)

3 - Normal (bottom-to-top)

4 - DeMorgan (bottom-to-top)

5 - Normal (right-to-left)

6 - DeMorgan (right-to-left)

7 - Normal (top-to-bottom)

8 - DeMorgan (top-to-bottom)

For example to call a right-to-left orientation B gate for a 74LS08 enter at the **DRAW-OBJECT** Text Window:

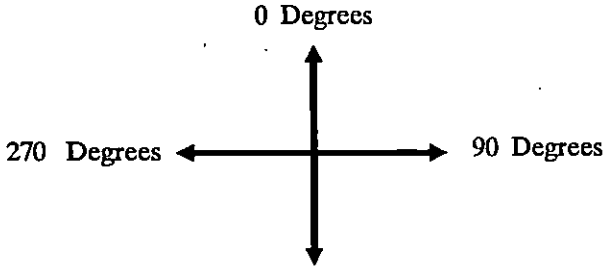
74LS08\5B <Enter>

VERTICAL TEXT

Proceed the text string with a backslash, '\' then a **<Space >** to vertically orient. You can also enter a vertical spacing parameter. For example, **\16 <space> ABC <Enter>** will space the letters A, B, and C 16 pixels vertically from each other. This is useful in creating text for pins of a part in the Object Editor.

CIRCLE/ARCS

First you must select either **NOTES-CIRCLE** or **BODY-CIRCLE.** For circles, type the radius in pixels then press **<Enter>** and place it on into the Drawing Window. Arcs can be created at any one degree increments. For example, type at the Text Window: **radius, begin degree, end degree** The following figure is the Cartesian plane that you should use as reference.



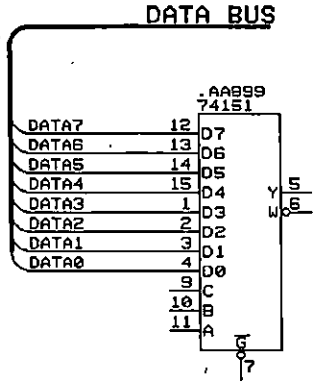
3

SCHEMA

CARTESIAN

BUS USAGE

Thick wires for buses carry no electrical connectivity. Buses are cosmetic and improve schematic legibility by reducing the amount of visual information in the drawing. The electrical connectivity of the schematic is carried by the labels on the wires. Example:



DATA0 through **DATA7** labels carry the connectivity to other wires that have the same labels.

USING PUSH / POP

The **FILE - PUSH** and **POP NN** commands can be used to help the user manage his hierarchical drawings, to assign a non-electrical schematic to a part , or to assign a ASCII Text file to a part.

PUSH

This command is used to put a schematic onto the **SCHDRAW** file stack. PUSH will automatically recognize a hierarchical object if it is created as per the specifications of the Manual. When the hierarchical object is selected by placing the cursor over the object and pressing the left mouse button or **<Ins>**, the Text Window appears with the file name for the hierarchical schematic. If **<Enter >** is pressed, then the hierarchical schematic will be loaded into the Drawing Window.

Non-electrical schematics or Text files can be assigned to a part by placing the filename into **PARM4** in the **DRAW-OBJECT** Parameter Line. For example:

If you created a PAL and you would like to have a schematic that shows the glue logic , but is not recognized by SCHPOST, then you might create a glue logic schematic called, GLUE.P01. Then when entering the PAL into the schematic type:

PAL16L8, U1, , , ,GLUE.P01

Using the same example, let's say that we want to assign a PLD input file that was created in an ASCII Text format, and the name of the file is PAL16L8.PLD. Then enter the part in the schematic by typing:

PAL16L8, U1, , , ,PAL16L8.PLD

Supplied with the software is a document object that will allow you to place multiple references to a schematic or text for documenting your schematics. This part could be used as follows:

DOCUMENT, , Manufacturing, Notes, , MANUFAC.TXT

Manufacturing
Notes

Note: Here the Reference designator was not used. However, be sure to include all the appropriate commas to delimit the input string.

POP NN

This command will let you increment through all of the files that are on the stack and ascend into a higher level of your hierarchical schematics, or to return from viewing a ASCII Text file. The NN represents the level of the stack and the number of files it contains.

CREATE A PART

SCHEMA has three distinct categories of Library Parts: **NESTED** Object, **BODY** Object, and **NAME** Object. The **NESTED** Object is a building block of the **BODY** Object. For example, the **NESTED** Object, **OAND** is an AND gate symbol with no pins attached. It's used by **BODY** Objects: T08-1A, T08-1B, T08-1C, and T08-1D.

Many different **NAME** Objects share the same schematic symbol. For example: 74LS08, 74S08, and 74F08 share the same **BODY** objects. The **NAME** Objects have up to four Decription Fields, and also contain a **BODY Field**, which is a maximum five character prefix taken from the **BODY** Objects to which it points. For example, the **BODY Field** for a 74LS08 is **T08**. *DO NOT INCLUDE A -1A EXTENSION*. The T08 **BODY Field** signifies that the **BODY** Objects are T08-1A, T08-1B, T08-1C, etc. can be called by the **NAME** Object, 74LS08.

4

PART CREATION PROCEDURE

I. Create the **BODY** Object

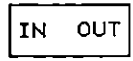
a.) Enter the Object Editor and make sure the Cursor Grid is ON. This insures that the pin definitions will be placed on grid.

b.) Place the text on the screen, which is to be the internal text of the part:

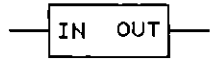
```
IN  OUT
```

This helps to avoid wasting space by determining the neccesary amount of space for the part.

c.) Next use **BODY -LINE** to outline the boundary of the part:



d.) Next select **BODY-HDR** and place the pin headers:



e.) Then use **BODY-DF PI** to define the pin definitions. Be sure to begin the pin definition with an attribute of :

I - Input

O - Output

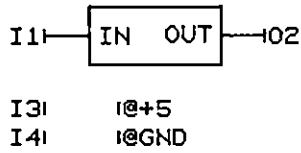
B - Bi-directional

T - Tri-state

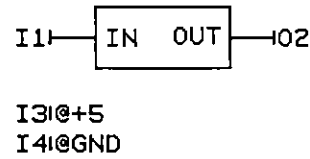
X - Don't care (*pin not involved in DRC*)

next characters are a signal name)

Also, assign any hidden power and ground definitions, if any. This is done by creating a pin definition for **I3** and **I4** facing to the right and then a pin definition using the '@' attribute for both **+5** and **GND**.



f.) Then move the two pairs of pin definitions so that the two headers overlap and appear as one.



g.) Save the **BODY** Object. Use **LIBR-SAVE** type the name, for example, **TEST-1A.**

II. Create the **NAME** Object

a.) Clear the screen using **LIBR-NEW.**

b.) Select **NAME-DESC1.** Enter a part number, and place.

c.) Select **NAME-DESC2.** Enter a description line for the part.

d.) Add any other decription information to **DESC3** and **DESC4.**

e.) Select **NAME-BODY.** Give it a **BODY** Field name with no extension. In keeping with our example, give the name, **TEST** (no extension). Maximum number of letters is five.

f.) Save the **NAME** Object. Select **LIBR-SAVE** and for example, type:

74TEST.

III. Invoke the Part to the Schematic

a.) Select **LIBR-ED SCH** and return to the Schematic Editor.

b.) Select **DRAW-OBJECT** and type: **74TEST.** Press and hold the mouse button, and place the part.

c.) If for example, you had a 'B' gate of **TEST** that you wanted to place (**TEST-1B**), then you would type: **74TEST\1B** , or use the **DRAW-GATE** command.

SCHPOST

Post-processing takes the graphical schematic pages that you have generated and outputs them to **Bill of Materials, Design Rule Checks,** and **Netlists.** When invoking, **SCHPOST** there are switches that can be entered on the command line so that **SCHPOST** doesn't bother loading the menu of options. Using the example of the ten page schematic, **PROJECTX,** let's say we wish to do a Design Rule Check. Then execute at the command line of DOS or the Manager :

SCHPOST PROJECTX D <Enter>

These are the switches and their use:

A (All post-processors, except the **SCHEMA-PCB** Pinlist)

B (Bill of Materials)

D (Design Rule Check)

N (Net List)

P (Pin List)

C (Net/Pin List Conversion)

R (Reports)

L (Lotus BOM Output file)

G (Dbase BOM Output file)

I (Delimited BOM Output file)

J (SDF BOM Output file)

S (SCHEMA-PCB Pin List)

W (Wire List)

RE-DIRECTING OUTPUT TO A FILE

SCHPOST automatically redirects the **Net List** (.NET), **Pin List** (.NPL), **Dbase** (.DBF), **Lotus** (.WKS), **Delimited** (.TXT), and **Standard Delimited** (.SDF) to files. However, if you wish to re-direct a Bill of Materials to a file then you would execute from the command line:

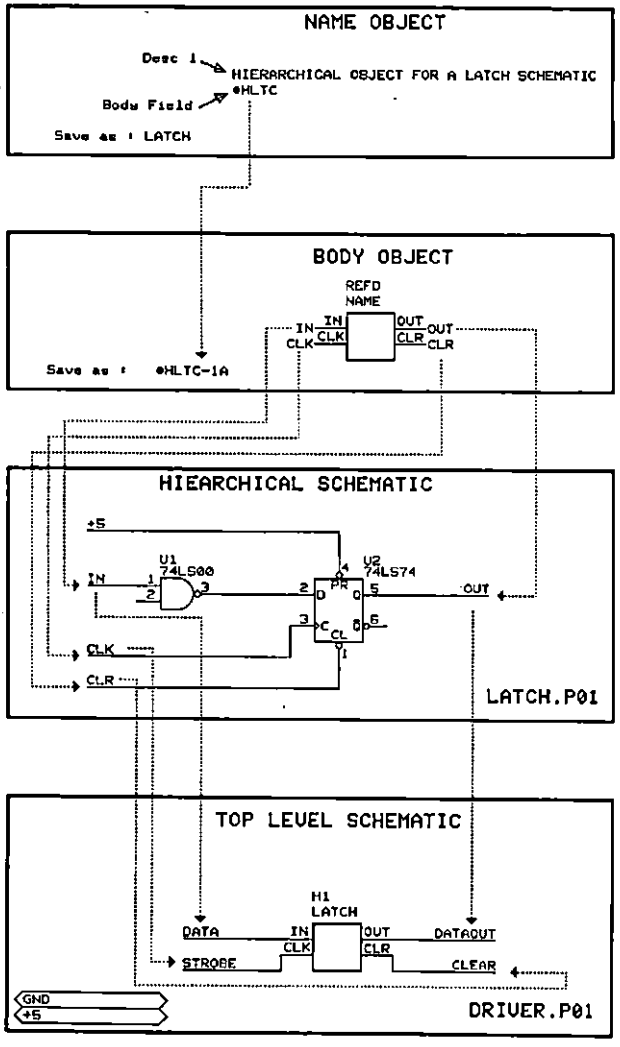
SCHPOST PROJECTX B >PROJECTX.BOM <Enter>

Be sure to include the DOS re-direct symbol '**>**' before the destination file name.

HIERARCHY

Hierarchy allows you to create a part that references a schematic page or pages. The schematic page that you wish to have as a hierarchical object will have a **NAME** Object in a **SCHEMA** library with the same name, minus the page extension. The **Body Field** of the **NAME** Object will be five characters as before, except that the first two characters must be "**#H**". The **Body Field** references a **BODY** Object. The **BODY** Object will have the net name labels as the pin definitions of the part. These label pin definitions connect to equivalent labels in the schematic that you are going to reference as a hierarchical object. This **BODY** Object must be saved with a name that begins with a '**#H**' and ends with an extension like, "**-1A**". The **GLOBAL** object can be used to define certain nets as global and not brought through the hierarchy process. An example would be **+5** and **GND**, or a net you do not wish to bring up into a higher level of schematic.

Note: The **FILE-PUSH** and **FILE-POP NN** can be used in **SCHDRAW** to assist in the use of Hierarchy. See the *DRAWING* section of this Guide.


NAME OBJECT
Desc 1
HIERARCHICAL OBJECT FOR A LATCH SCHEMATIC
Body Field
•HLTC
Save as : LATCH
BODY OBJECT
REFD
NAME
IN
CLK
OUT
CLR
Save as : •HLTC-1A
HIEARCHICAL SCHEMATIC
+5
U1
74LS00
U2
74LS74
IN
OUT
CLK
CLR
PR
CL
LATCH.P01
TOP LEVEL SCHEMATIC
H1
LATCH
DATA
STROBE
DATAOUT
CLEAR
GND
+5
DRIVER.P01

5

LIBRARY PHILOSOPHY

We suggest that you do not add any new parts to our 9 Libraries. You should create a User library to store any new parts that you create. You need only back up this library if something happens to your Hard drive. Then you can re-install **SCHEMA** and it's libraries with no data loss.

Create User Library

I. Invoke **SCHLIB**, Library Manager

II. Select **CREATE LIBRARY** 2

III. **SCHLIB** will ask you what size: **S**-Small, **M**-Medium, or **L**-Large. Type the letter corresponding to the needed size.

IV. **SCHLIB** will then prompt you for a the name, path, and extension. Give the drive and path to where you want the library. Then give a name within eight characters and a **.LBR** extension. For Example:

6

C:\SCHEMA\LIB\MYPARTS.LBR

ASSIGN A USER LIBRARY

We suggest that you make your new library the default library, **LIBR1** in the **DRAWING.CFG**. Where **DRAWING** represents the name of the schematic. Thus, when using the Object Editor in **SCHDRAW** and you save a part, it will automatically be saved to your library in the #1 library position. Use the **SCHCFG** program to place your new library into the **LIBR1** position of your **DRAWING.CFG** file. Also, you will want to place your new library into the **LIBR1** position of the **SCHEMA.CFG** file, because the **SCHEMA.CFG** is the template file from which all of the **DRAWING.CFG** files are created. Therefore, all of the new projects you begin will have your new library always assigned to **LIBR1**.

ARCHIVE A LIBRARY

Let's say that you have just created a ten page schematic project, **PROJECTX.P01** through **PROJECTX.P10**. However, now you wish to send it to a collegue or store it away, but you don't want to save away all the libraries with it. This is when you should localize your libraries using the program, **SCHLOCAL**.

I. Use **SCHLIB** to create a new library. For example: **PROJECTX.LBR**.

II. Place **PROJECTX.LBR** in the **LIBR1** position of the **PROJECTX.CFG**.

III. Execute: **SCHLOCAL PROJECTX**

IV. Now all the parts used in the schematic pages for **PROJECTX** are copied to the **PROJECTX.LBR**. You can now enter the **PROJECTX.CFG** and nullify **LIBR2** through **LIBR9**, and use only the new library in the **LIBR1** position.

SCHLIB

USAGE: **SCHLIB <Enter>**

SCHLIB is the **SCHEMA Library Manager**. The **LIB1** and **LIB2** variables have NOTHING in common with **LIBR1** and **LIBR2** found in the .CFG files. The commands are:

SWAP LIBRARIES 1 AND 2 - Swaps the **LIB1** and **LIB2** variables.

STATUS LIBRARY 1 OR 2 - Reflects : Total Library size, Total Recoverable bytes, In Use directories, Number of Recoverable directories, and Number of Free directories.

CREATE LIBRARY 2 - Used to create a library. There are three sizes: Small, Medium, and Large.

COMPRESS LIBRARY 1 - Recovers the directories that were used but deleted or overwritten.

DIRECTORY OF OBJECTS LIBRARY 1 - To display or **PRINT** a library directory. When selected it will prompt you for a Search String.

Examples:

74LS08 (Displays the 74LS08 part name)

74* (Displays a list of all objects beginning with 74)

*** <space> ** (redirects a list of the whole directory to the printer)

*** <space> \SCHEMA\LIB1.PRT** (redirects the list of the directory to a file called **LIB1.PRT** to the **\SCHEMA** directory)

COPY OBJECT LIBRARY 1 TO 2 - Copies an object or objects from **LIB1** to **LIB2.**

ERASE OBJECT LIBRARY 1 - Deletes an object or objects from LIB1.

EDIT OBJECT LIBRARY 1 - Edits the Name Objects for LIB1.

ASCII IN/OUT NAME OBJECTS LIBRARY 1 - Allows the user to ASCII IN or OUT Name Objects to and from the library assigned to LIB1. The format of the ASCII Syntax is:

[

NAME

DESC 1

DESC 2

DESC 3

DESC 4

BODY

]

Be sure to enter a blank space even if one of the fields are not to be used. The ASCII file can be read into SCHLIB and any errors will be re-directed to an error file. The error file is set up such that you could modify it and then read the error file back in to complete the ASCII IN process. All information between the closed and open brackets are ignored. When modifying the Name Objects make sure that you are using a non-document text editor.

QUIT - Quit **SCHLIB**.

SCHCFG

There are two different configuration files. One is the main configuration file, **SCHEMA.CFG**. Second is the **DRAWING.CFG** file, where **DRAWING** is the name of the schematic project. The **SCHEMA.CFG** file is located in the **SCHEMA** directory with the .EXE files. This is the template file from which the **DRAWING.CFG** files are created. The **DRAWING.CFG** takes presidence over the **SCHEMA.CFG**. However, GRAPHICS and PRINTER driver information is always read from the **SCHEMA.CFG** file. The **SCHCFG** program allows access to these files.

USAGE : **SCHCFG PROJECTX <Enter>** (*for PROJECTX.CFG*)

or

SCHCFG SCHEMA <Enter> (*for SCHEMA.CFG*)

The following options are displayed:

HARDWARE CONFIGURATION - Configures the graphics, printer, and plotter devices.

DRAWING PARAMETERS CONFIGURATION - Configures the panning speed, the repeat coordinates, CGRID, VGRID, wire headers, and diagonal lines.

DRAWING TITLE BLOCK CONFIGURATION - Configures the Title block information of the drawing.

LIBRARIES CONFIGURATION - Configures the component libraries to be used with a drawing.

BOM FORMAT CONFIGURATION - Configures the Bill of Materials header and output fields.

LOAD CONFIGURATION - Loads another configuration file.

SAVE CONFIGURATION - Saves the current configuration file.

QUIT - Returns to the Manager or DOS.

SCHPRINT

USAGE :

SCHPRINT <Space> DRAWING <Space> OPTIONS <Enter>

DRAWING represents the schematic you wish to be printed, and **OPTIONS** are the command line switches for varied output. Remember that to alter the command line in the MANAGER to press **<Space>** first.

DRAWING parameters can changed to:

DRAWING - prints all pages in the schematic

DRAWING.P01 - prints the first page.

DRAWING.P01,P07,P13 - prints pages 1, 7, and 13.

OPTIONS parameters can be one or multiple:

/M - specifies the Modes of the printer. Where **M** could be A,C,D,etc. depending on which printer you have see *APPENDIX E*. Default is A.

/N - specifies which ZOOM level to print the page: **1** - 1:1 (Default), **2** - 2:1, **4** - 4:1, and **8** - 8:1

/O - (*letter 'O'*) specifies to output the print to a file. For example: PROJECTX.P01 will be re-directed to PROJECTX.T01. The output can be directed to another directory as well. For example: **/O=A:\PRINT**. If you wish to ouptut to a different device then type : **/O=LPT2** or **/O=COM1**

/R - Rotates the printout 90 degrees.

WHAT IS THE SIZE OF THE PRINTOUT

1. Check *Appendix E,* for the printer driver and mode you are using.

2. Find out how many pixels are in the X and Y directions of a page by using **SCHPAGE**. Execute : **SCHPAGE PROJECTX.P01.**

3. Depending on the mode there is a DPI (dots per inch). Each page pixel is mapped to one dot. Calculate.

For Example: A mode for an EPSON printer is 120 x 108 dpi. The page to print is 1600 x 1216 pixels on narrow carriage paper.

Width = (8")(120 dpi) = 960 pixels

Length = (10")(108 dpi) = 1080 pixels This will not print on one page, but four.

Use **SCHPAGE** to make a custom page if the supplied pages don't meet your needs.

BACKGROUND PRINTING

To print a schematic page in the background mode, you must invoke the DOS, **PRINT.COM** print spooler. This is done by typing at the DOS prompt:

PRINT <Enter>

Then **PRINT.COM** asks which device. The default is **PRN**, so you can just press **<Enter>** again. Then invoke **SCHDRAW** and select **FILE-PRINT.** The Text Window will show the schematic name with a 'T' extension. Press **<Enter>** and the print is re-directed to disk file with the 'T' extension. Then the print utility will begin printing in background mode.

SCHPLOT

SCHPLOT outputs **HI, HP, Calcomp, DXF,** and **PostScript** outputs.

USAGE:

SCHPLOT <space> DRAWING <space> OPTIONS <Enter>

Where **DRAWING** is the schematic you wish to be plotted and **OPTIONS** are the command line switches for varied output. Remember that to alter the command line in the MANAGER to press **<space>** first.

DRAWING parameters can be changed to:

DRAWING - plots all pages in the schematic

DRAWING.P01 - plots the first page.

DRAWING.P01,P07,P13 - plots pages 1, 7, and 13.

OPTIONS parameters can be one or multiple:

/C - Prompts to change paper between each plot

/E - Issues an eject command at the end of the plot, if supported.

/O - (letter 'O') specifies to output the plot to a file. For example: PROJECTX.**P01** will be re-directed to PROJECTX.**S01**. The output can be directed to another directory as well. For example: **/O=A:\PLOT.** If you wish to output to a different device the type: **/O=LPT2 or /O=COM1**

/P - Cause the parts to be defined to the plot prior to drawing the parts. Supported for DXF only.

/R - Rotates the printout 90 degrees.

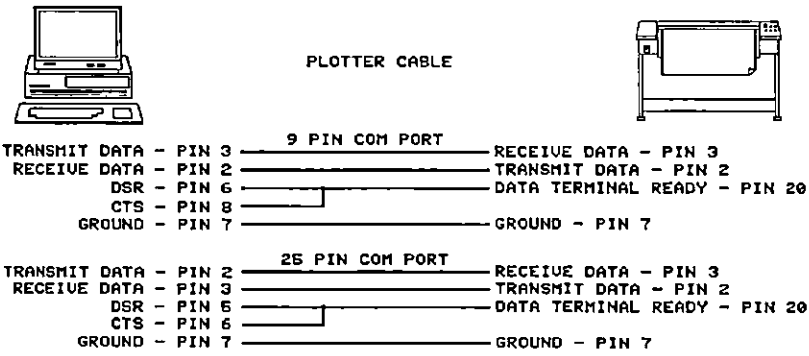
FACTS:

SCHPLOT looks first in the **DRAWING.CFG** file for the plotter driver and plotting parameters, defaulting to the **SCHEMA.CFG** file only when the **DRAWING.CFG** file is not found.

The **MODE** command MUST be invoked in order for **SCHEMA** to plot. Make sure there is not more than one **MODE** command in your AUTOEXEC.BAT that could conflict with each other.

Example: **MODE COM2:9600,N,8,1,P <Enter>**

There are two methods for which to plot, hardware-handshaking and software-handshaking. **SCHEMA** uses HARDWARE-HANDSHAKING. Just because your plotter works with a program like **AutoCAD**, does not mean it works with **SCHEMA**. **AutoCAD** uses software-handshaking, or Xon/Xoff. **SCHEMA** requires an extra hardware line called the **DTR**(Data Terminal Ready) on the plotter to direct the computer when to start and stop the transmission of data. The HARDWARE-HANDSHAKING


PLOTTER CABLE
9 PIN COM PORT
TRANSMIT DATA - PIN 3
RECEIVE DATA - PIN 2
DSR - PIN 6
CTS - PIN 8
GROUND - PIN 7
RECEIVE DATA - PIN 3
TRANSMIT DATA - PIN 2
DATA TERMINAL READY - PIN 20
GROUND - PIN 7
25 PIN COM PORT
TRANSMIT DATA - PIN 2
RECEIVE DATA - PIN 3
DSR - PIN 6
CTS - PIN 6
GROUND - PIN 7
RECEIVE DATA - PIN 3
TRANSMIT DATA - PIN 2
DATA TERMINAL READY - PIN 20
GROUND - PIN 7

cable must be configured as specified below in order for **SCHEMA** to plot. **AutoCAD** will work with this cable as well.

SCHPLOT requires you to give a Plot X Length and Plot Y Length that represents the plot size. These parameters are entered into the **DRAWING.CFG**. There are also two offset parameters: **PXOFF** an **PYOFF**. These parameters determine where on the sheet the drawing will be plotted.

NOTE: **HP** plotters' origins are always located at the center on the paper. Thus, if you wish to force the plot to the lower left hand corner, negative offsets will have to be given. For example: Let's say that we are plotting to a B size sheet, and the maximum plotting area for this paper is 16000 mils by 10000 mils. The **PXLEN** is 16000 and the **PYLEN** is 10000. Therfore, to force the plot to the lower left hand corner the offsets would have to be: **PXOFF** = -8000 and **PYOFF** = -5000.

If you are generating a **DXF** or **PostScript** output then you will need to add the "/O" (*letter O*) switch to the command line to tell **SCHPLOT** to redirect the plots to files. The plot files will have an 'S' extension. For Example, the .**P01** drawing plot will be sent to .**S01**.

If you wish to create a Drawing page that completely fills a certain plotter sheet, then review page 8-15, *Customizing Pages For a Particular Plotter.*

SCHPAGE

USAGE: **SCHPAGE <Enter>**

SCHPAGE is used to Create a new page or display the size of an existing schematic.

BLANK - LANDSCAPE

PPAGE1.P99 - Virtual C-size (1024 x 960 pixels)

PPAGE2.P99 - Virtual D-size (1600 x 1216 pixels)

PPAGE3.P99 - Virtual E-size (1984 x 1472 pixels)

PPAGEA.P99 - Standard A-size (640 x 480 pixels)

PPAGEB.P99 - Standard B-size (992 x 640 pixels)

PPAGEC.P99 - Standard C-size (1312 x 992 pixels)

PPAGED.P99 - Standard D-size (2016 x 1312 pixels)

PPAGEE.P99 - Standard E-size (2624 x 2016 pixels) 662K Expanded memory required

BLANK - PORTRAIT

PPPAGE1.P99 - Virtual C-size (960 x 1024 pixels)

PPPAGE2.P99 - Virtual D-size (1216 x 1600 pixels)

PPPAGE3.P99 - Virtual E-size (1472 x 1984 pixels)

PPPAGEA.P99 - Standard A-size (480 x 640 pixels)

PPPAGEB.P99 - Standard B-size (640 x 992 pixels)

PPPAGEC.P99 - Standard C-size (992 x 1312 pixels)

PPPAGED.P99 - Standard D-size (1312 x 2016 pixels)

PPPAGEE.P99 - Standard E-size (2016 x 2624 pixels) 662K Expanded memory required.

BORDERED - LANDSCAPE

BPAGE1.P99 - Virtual C-size (1024 x 960 pixels)

BPAGE2.P99 - Virtual D-size (1600 x 1216 pixels)

PAGE3.P99 - Virtual E-size (1984 x 1472 pixels)

BPAGEA.P99 - Standard A-size (640 x 480 pixels)

BPAGEB.P99 - Standard B-size (992 x 640 pixels)

BPAGEC.P99 - Standard C-size (1312 x 992 pixels)

BPAGED.P99 - Standard D-size (2016 x 1312 pixels)

BPAGEE.P99 - Standard E-size (2624 x 2016 pixels) 662K Expanded memory required.

The Virtual pages (1,2,3) are smaller and due fit the whole schematic on one sheet of paper. For example , **BPAGE2.P99** fits one A-size sheet on a HPLaserJet. The Standard pages will be true size for a 60 dpi printer.

MEMORY LIMITS

The executable and buffers use up **170K** of RAM of what's left of the Standard 640K DOS memory. The Image buffer takes up the rest. For example, we wish to load a page created from **BPAGED.P99**. The size of this page is **2016** x **1312** pixels. Use the following equation:

(2016 * 1312) / 8 = 330,624 bytes

TOTAL: 170,000 + 330,624 = 500,624 bytes

Execute the DOS command, **CHKDSK** to see how much free RAM is available on your system. If it is greater than **501K** then the page can be loaded.

CUSTOMIZING A TITLE BLOCK

The template pages (**.P99**) are all legitimate pages that can be loaded into **SCHDRAW** and altered to reflect your Title block information. The procedure would be as follows:

1. Enter **SCHDRAW** and select **FILE-LOAD** and load a **.P99** file of your choice.

2. Alter the Title block. Erase the **OMATION** logo. Create one of your own and replace it.

3. Add additional automatic Title Block parameters that will be loaded from the .CFG file. Let's say we want to add an unused parameter like number **8** from the **TITLE BLOCK MENU** of the **DRAWING.CFG.** Use **NOTES-TEXT** and type: **<F8>** (Include the brackets)

Place the parameter in the Title Block. This is now a pointer to load whatever string of text that resides in parameter 8 in the **DRAWING.CFG.** See **NOTES-TEXT** in *Chapter 3, SCHDRAW* for more details.

4. After making the changes use **FILE-SAVE** to save the page to itself or another file name with a **.P99** extension.

CUSTOMIZING A PAGE FOR PLOTTING OR PRINTING

A page in **SCHEMA** is relative to the printer used or to the size sheet of paper to plot. **SCHEMA** has different printers with different dot per inch modes that affect how large the printout will be. Also **SCHEMA** can scale a plot to any size. Thus, the supplied template pages can be used as a model from which to create a page that matches your schematic needs for your printer or plotter. See *Chapter 8, Utilities* pages **14** and **15** for details on customization.

CHANGING PAGE SIZES

If you are running out of room in your schematic and you wish to take your schematic to a larger page then use **EDIT-REGION-SAVE** to draw a region around the schematic and save it to disk. When you enter the new page and select **EDIT-REGION-LOAD**, there is a **NEWREF** command light that will automatically re-assign new reference designators to the region. If you wish to retain the original reference designators then turn this command light off before you load the region.

CONNECTIVITY TO THIRD PARTY SOFTWARE

NET and PIN List CONVERSION

Within **SCHPOST** there is an option for **NET/PIN LIST CONVERSION.** This selection will ask if you wish to overwrite any existing **SCHEMA** netlist and pinlist already created. Answer 'Y' and new **.NET** and **.NPL** files will be created on the hard drive. Then **SCHPOST** will invoke the program, **CONVERT.EXE** and a list of third party netlist formats appear. Select the output you want created. The list is as follows:

Cadnetix/SCI-CARDs Net List

Calay Net List

CBDS Net List

Computer Vision Net List

dataCon Wire-Wrapping Net List

Visionics (EE Designer) Net List

FutureNet Net List

FutureNet Pin List

Intergraph Net List

P-CAD PDIF Net List

Racal Redac Net List

PSpice Net List

Telesis Net List

AutoCAD (DXF Format)

DXF output is built into the **SCHPLOT** program, which gives the user the ability to scale the output. The procedure is as follows:

1. Let's say we have a schematic called, **PROJECTX**. We want to take the first page to **AutoCAD**. Use **SCHCFG** to load the config file **PROJECTX.CFG** and make sure the plotter is setup for **DXF**. Check the X and Y lengths for scaling.

2. Use **SCHPLOT** to execute the following statement :

SCHPLOT PROJECTX.P01 /O /P <Enter>

The **/O** switch is required and redirects the **DXF** output to the file **PROJECTX.S01.**

The **/P** switch is optional and will group the schematic parts into BLOCKS in **DXF.**

3. You must rename the file **PROJECTX.S01** to **PROJECTX.DXF** for **AutoCAD** input. Copy the input file to a directory of your choice.

4. Invoke **AutoCAD** and select the option, **DXFIN**. Read the **PROJECTX.DXF** file into **AutoCAD.**

PostScript Output

PostScript output is built into the **SCHPLOT** program, which gives the user the ability to scale the output. The procedure is as follows:

1. Let's say we have a schematic called, **PROJECTX**. We want to take the first page to **PostScript**. Use **SCHCFG** to load the config file **PROJECTX.CFG** and make sure the plotter is setup for **PS**. Check the X and Y lengths for scaling.

2. Use **SCHPLOT** to execute the following statement :

SCHPLOT PROJECTX.P01 /O /R <Enter>

The /O switch is optional and redirects the **PostScript** output to the file **PROJECTX.S01.**

The **/R** switch is optional and will rotate the schematic 90 degrees. When this option is specified the origin is at the lower left corner of the paper. X-coordinates increase left to right and Y-coordinates increase bottom to top.

3. You must have have a **PostScript** language interpreter installed as part of your printer or a third party software interpreter. **NOTE:** The file is output just like a plotter. Therefore, if you are sending the file through a COM port you must use the same cable as specified for the Pen plotters.

(see *PLOTTING* of this manual)

TIFF

This output format is created from the **TIFF.DRV**, printer driver. This format can be read by **PageMaker, Word Perfect, Ventura,** and others. The procedure is as follows:

1. Let's say we have a schematic called, **PROJECTX.** We want to take the first page to **TIFF.** Use **SCHCFG** to load the config file **SCHEMA.CFG** and make sure the printer is setup for **TIFF.DRV.**

2. Use **SCHPRINT** to execute the following statement :

SCHPRINT PROJECTX.P01 /O /R <Enter>

The /O switch is required and redirects the TIFF output to the file **PROJECTX.T01.**

The /R switch rotates the output 90 degrees. For some desktop publishing software like Ventura this switch is needed to have the drawing displayed in portrait style on the page.

All other switches supported by **SCHPRINT** can be used as well.

3. Load your favorite desktop publishing program and input the TIFF file per your word processing program's specifications.

NOTE : If you wish to cut a region to output to a file, use **EDIT-REGION-SAVE** and give the region an extension of **(.P01)**.

ECO to SCHEMA-PCB

Foward and Back annotation are useful tools for integration of schematic and PCB board software, thus a better understanding of the process aides in the productivity of the tools. First, let's review exactly what happens when integrating **SCHEMA** and **SCHEMA-PCB.**

A schematic called **PROJECTX** has been completed. The Post-processor, **SCHPOST** is invoked, and the **SCHEMA-PCB PIN LIST** option is chosen. This option should only have to be selected once per design. **SCHPOST** will then create two files: **PROJECTX.SCH,** which is a FutureNet Netlist and **PROJECTX.FWR,** which is a **SCHEMA** pinlist. Both of these files are copied to the **\PCB\FILES** directory on the same Hard drive as **SCHEMA.**

Next, **SCHEMA-PCB** is invoked and the command **In\Out - SCHEMA In** is selected and given the name **PROJECTX** to input. **SCHEMA-PCB** then reads the **PROJECTX.SCH** file into the system and checks for errors. If there are no errors found then the message, **"NO ERRORS FOUND",** appears and **SCHEMA-PCB** renames the file **PROJECTX.FWR** to **PROJECTX.$$$.** This **.$$$** file represents the net names and nodes of the PCB job at this point in time.

Now let's say that your Engineering Dept. has altered the design and you have updated your schematic accordingly. Then we will select the **SCHECO** command and choose the **"F"** option. **SCHECO** then generates a new **PROJECTX.FWR** and **PROJECTX.SCH** and then compares the new **PROJECTX.FWR** to the **PROJECTX.$$$.** Any differences in the files is then reflected in a **PROJECTX.ECO** file. **SCHEMA-PCB** is invoked and the current job is loaded.

Then the command **ECO - Frm SCH** is selected, and asks for a file to read. The file, **PROJECTX** is typed and PCB reads the **PROJECTX.ECO** file into the PCB board database. If there are **"NO ERRORS FOUND"** while reading the **PROJECTX.ECO,** then the **PROJECTX.$$$** is renamed to **PROJECTX.$BK** and the new **PROJECTX.FWR** is renamed to **PROJECTX.$$$.** Thus upon successful ECO the **.$$$** is updated to reflect the net names and nodes of the new PCB board database. If there had been errors then the **.$$$** file would have stayed the same. Therefore, you can see the importance of maintaining the integrity of the **.$$$** file.

If you want to Back annotate gate swapping or reference designator changes, then at the prompt at either the PCB command, **ECO - To Sch - Rename Part** or the **Place - AutoPlace - Swap Items - Swap Gates** , enter the file name **PROJECTX.BCK.** Then when SCHECO is invoked and the "**B**" option is chosen a **PROJECTX.REF** is created using the information found in the **PROJECTX.BCK** file. The **PROJECTX.REF** is used to update the schematic.

Be aware that if you swap gates or change reference designators that you must do the Back Annotation before you try to do any Foward Annotation. Also, if you have already completed a Back Annotation (running **SCHECO**) once before, but you have realized that you did not fully complete the task, remember to erase the current **.BCK**. Thus, you create a new file so that you do not Back Annotate the changes that you have already input to the schematic.

If in the event that the **PROJECTX.$$$** file was somehow destroyed, you can regenerate this file. First, get the schematic back to the point at which it should match the PCB job database. (*SCHEMA automatically creates backup files of the schematic.*) Then select the **SCHEMA-PCB PIN LIST** option in **SCHPOST** again. This will make the **.FWR** and the **.SCH** files. Then invoke PCB and load the current job. Use the **Check - SCHEMA Chk** command to compare the **.SCH** file against the job database. If the CHECK returns the message, "**NO ERRORS FOUND**", then you can use DOS to manually rename the **.FWR** to a **.$$$** file. Thus, the schematic and the PCB job are synchronized, and the new schematic can be loaded and the Forward Annotation executed.

See pages in the *SCHEMA-PCB Reference Manual* ,1-110 - 1-112, 6-21, 9-1 - 9-3, 9-9, & 9-18.

See *SCHECO, Chapter 8* in the *SCHEMA III Reference Manual.*

801 Presidential Drive

Richardson, Texas 75081

(214) 231-5167

FAX (214) 783-9072

BBS (214) 231-1350

Copyright OMATION, Inc. 1990 OMATION, the OMATION Logo, and SCHEMA III are trademarks of OMATION

All other third party trademarks are referenced in the SCHEMAIII Reference Manual